KiCad PCBNew Primer
A quick recap, to refresh my KiCad PCB abilities

Summary: This document should be read before starting on a PCB layout with KiCad. These tips may apply to other PCB software as well.

Route Power First: Route power first, and pay attention to how many layers you will need. Depending on how many signals, and where they are going, you may need a 4 or higher layer board. 144LQFP was difficult using 100% of the pins, but possible on a 2 layer board.

Case and Mounting Holes: Put the mounting holes into the PCB early, to avoid rerouting layer. Same with the case.

Keyboard Shortcuts: Use keyboard shortcuts. X to choose 'add trace', PageUp and PageDown to cycle through layers, there is a hotkey for changing layer types. O will add a component (enter or left click after selecting O)

Measuring: There is a ruler in KiCad.

Ground Pours: If you want to do a ground pour that is split, there is the keepout area. This keepout area has an option for component or fill keepout. Choose the fill keepout. Test on a blank project to see workflow.

Via Size, Trace Size, Fab House Guidelines: Set the fab house guidelines into KiCad before starting, otherwise you will have to change items later. e.g. if you set the Vias to be too small, you will have to edit all vias later. There are templates available with DRC guidelines for OSHPark, as an example.

White Background: In 5.0.0rc2 and newer (assumed) change the background colour to white by clicking on the option. It may be greyed out, but still selects.

Board Edge: Edge.Cuts and use a 1 mil or .001 inch "add graphic lines" on the edge cut layer.

Spark Gap: Refer to spark gap radio for other tips on PCB design such as using pin headers instead of pads, and of breaking out all pins for future hacking of designs.

Modular: Make separate boards, and reuse what you can. Some things are trivial to layout, and don't need separate boards. Anything that is difficult to layout, consider using it's own board.

Net Names: Give net names to tracks. This will make identification easier on layout. E.g. see here how I've added labels to nets that might not need them.

Component Values: If you are building by hand, add component values to the footprints silk screen to make it easier to identify. See picture.
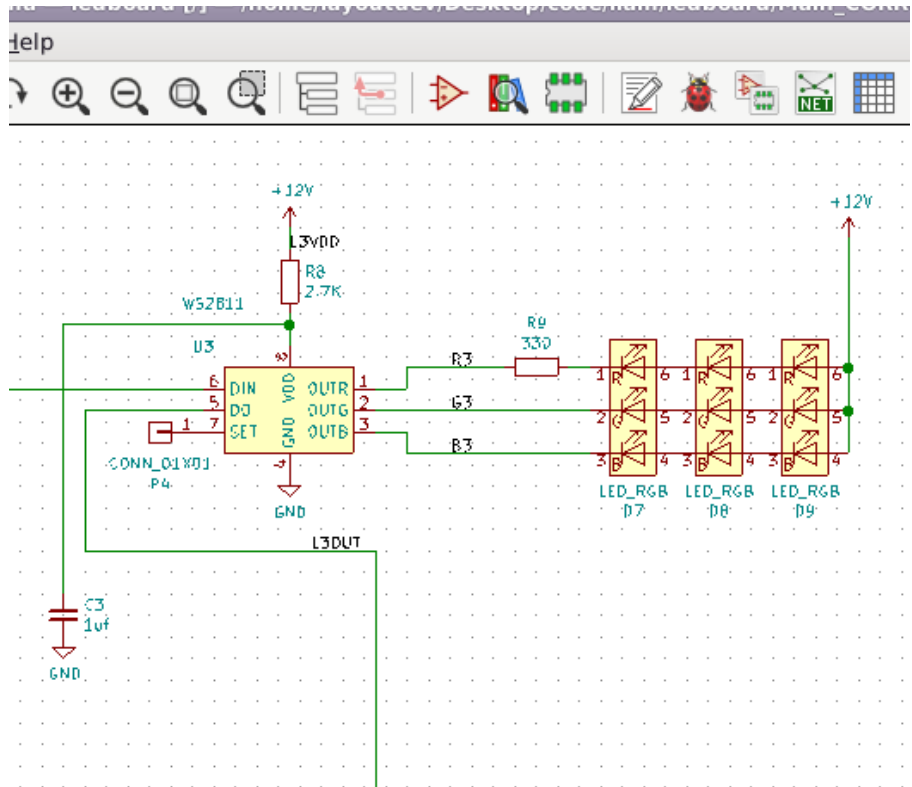
Adding net labels, makes line identification on layout easier. Here we have R3, G3, and B3, telling us that these leds are going to the input of the LED driver.
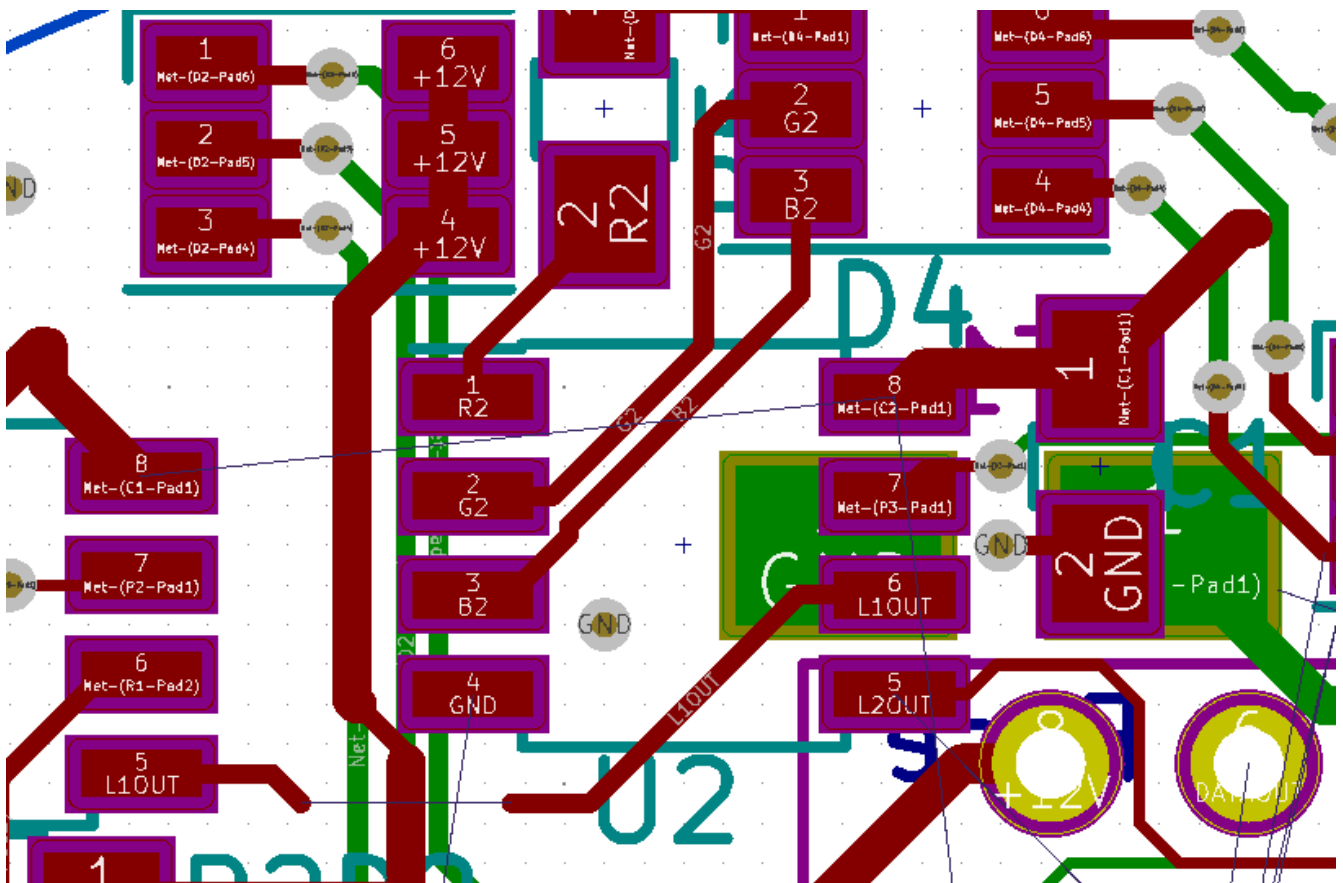
This converts the mystifying names of NET-(P2-Pad4) to something human readable like R2, G2, B2.
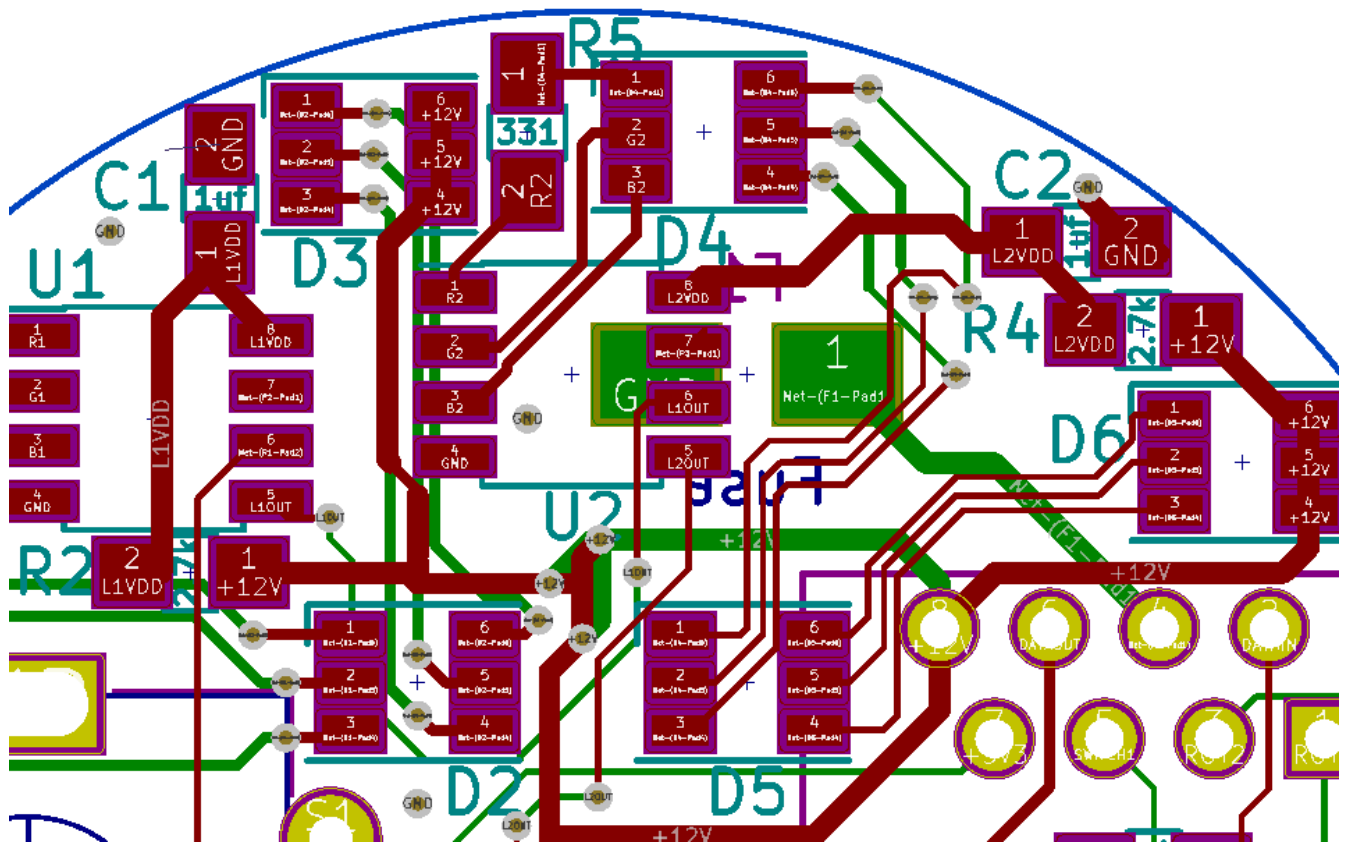
By adding a text of .02 size to 0805 footprints, you can easily fit within the provided kicad footprint. This makes soldering easier later. See here, labels such as 1uf, or 2.7k.

How to import pictures into pcb
Get a black and white picture. Make the part you want white. Make the background transparent. Save as png. Import into bitmap 2 component. Image should be a white + transparent image.

Related Resources: Dave Jones PCB guide, Spark Gap